Final Report for Period: 06/1999 - 05/2004
Principal Investigator: Sotiropoulos, Fotis .
Organization: GA Tech Res Corp - GIT
Title:
Career: Advanced Numerical Modeling of Bridge Foundation Scour

Submitted on: 08/31/2004
Award ID: 9875691

Project Participants

Senior Personnel

Name: Sotiropoulos, Fotis
Worked for more than 160 Hours: Yes
Contribution to Project:

Post-doc

Name: Jones, Casey
Worked for more than 160 Hours: Yes
Contribution to Project:
C. Jones is developing parallel algorithms for solving the unsteady, Navier-stokes equations in complex geometries.
Partly supported by NSF.

Graduate Student

Name: Ge, Liang
Worked for more than 160 Hours: Yes
Contribution to Project:
He is developing a 3-D unsteady numerical method for predicting large-scale unsteady vortices in real-life bridge foundation geometries.
Supported by NSF.

Name: Tang, Hansong
Worked for more than 160 Hours: No
Contribution to Project:
H. Tang has developed a domain decomposition method for solving the 3D, unsteady, Navier-Stokes equations in complex geometries. This code is the basic hydrodynamic model for simulating bridge foundation flows. H. Tang graduated with his Ph.D. in December 2001.

Name: Lackey, Tahirih
Worked for more than 160 Hours: Yes
Contribution to Project:
Ms. Lackey is developing Lagrangian tracking algorithms for simulating transport and stirring of tracers in vortex dominated flows. The main focus of her current research is on the chaotic advection of passive scalars in swirling flows with vortex breakdown. The numerical tools she is developing, however, will form the basis for the Lagrangian/Eulerian sediment transport model that will be developed in this research.

Undergraduate Student

Technician, Programmer

Other Participant

Research Experience for Undergraduates

Organizational Partners

Georgia Department of Transportation

GADOT is funding a project aimed at combining field measurements, laboratory experiments, and 3D numerical modeling to address scour related problems in the state of Georgia. Funds from this project have been used by the PI to obtain matching NSF funds for CMS-9875691.

U.S. Geological Survey

With funding from GADOT, USGS stuff will carry out field measurements at several bridge sites throughout the state of Georgia. These measurements will be used to calibrate and validate the 3D numerical model developed by the PI with NSF funding.

Other Collaborators or Contacts

I collaborated with Prof. Terry W. Sturm, School of Civil and environmental Engineering, Georgia Tech. With funding from GADOT, Prof. Sturm carried out a series of laboratory-scale scour experiments, which provided data for calibrating and validating the 3D numerical model that was developed with NSF funds.

Activities and Findings

Research and Education Activities:

Research activities

The following tasks were accomplished:

1. A domain-decomposition method for solving the 3D, unsteady, Navier-Stokes equations in arbitrarily complex geometries has been developed. The method employs a novel mass-flux based interpolation technique for interface boundary conditions, which allows coherent vortices generated in one subdomain to cross into adjacent subdomains without distortion. The method has been extensively validated by applying it to simulate a variety of steady and unsteady laminar flows in complex, multi-connected geometries. For all cases, the agreement with available data or previous numerical simulations has been found to be excellent.

2. A Lagrangian algorithm for calculating the paths of passive particles in 3D unsteady flows has been developed and incorporated in the domain-decomposition code. Lagrangian particle tracking is currently employed to calculate flow streaklines in order to visualize and elucidate the structure of the foundation-induced vortices. This algorithm will also form the basis for a Lagrangian/Eulerian approach for modeling sediment transport processes that lead to scour.

3. A novel algorithm has been developed for improving the performance of standard dual time stepping, artificial compressibility (AC) schemes in solutions of the 3D, unsteady, Navier-Stokes. The new algorithm is a Fractional-step Artificial Compressibility (FSAC) method, which combines features of pressure-based, projection algorithms with the standard dual time-stepping AC method. Its implementation has been shown to improve the efficiency of the standard dual time-stepping AC scheme by as much as 60 percent while preserving the second-order temporal accuracy of the original scheme. Significant efficiency gains are expected from the implementation of this algorithm in simulations of unsteady, bridge foundation flows.

4. A novel experimental technique has been developed for visualizing the 'footprints' at the free-surface of large-scale, unsteady vortices in the vicinity of bridge foundations. The method relies on digital photography and it is very simple to implement. It has already been applied to visualize large-scale, coherent vortices in the vicinity of several bridge abutments at the Georgia-Tech scour flume. The general vortical features revealed by these experiments are broadly consistent with those uncovered by numerical simulations of flows past similar geometries using the 3D, unsteady, Reynolds-averaged equations.

5. The domain decomposition code has been parallelized using the OpenMP library. The parallelized code runs approximately 10 times faster on a 16 processor Origin 2000 machine. Work is currently under way to develop a parallel version of the code using MPI.

6. The parallel domain decomposition code has been extended to turbulent flows. Steady and unsteady RANS calculations were carried out for various bridge sites in the state of Georgia. Note that the bridge geometry at these sites is very complex, typically involving multiple bundles of cylindrical and or rectangular piers with fenders and other features used for enhancing structural stability. Furthermore, we succedded in simulating the entire bridge section on the natural topography of the river. The domain decomposition code we have developed allows, for the first time, unsteady RANS simulations to be carried out for such complex geometries on very fine computational grids (typically 2-3 million points are used). Results were obtained for three bridge sites in the state of Georgia. All sites were characterized by extremely complex geometry with the pier bents typically consisting of multiple cylindrical or rectangular piers.

7. The unsteady RANS flow solver has been successfully validated using experimental measurements obtained at the Georgia Tech Hydraulics laboratory. The computed time-averaged velocity profiles were found to be in excellent agreement with the experimental data for all cases considered. The comparisons also showed that even a slight misallignment of the flow relative to the axis of the pier bent can have a profound effect on the structure of turbulence in the vicinity of the foundation.

Overall, these comparisons demonstrated that the numerical model developed in this work is the most advanced today hydrodynamic simulation tool capable of quantitatively accurate predictions of real-life hydraulic engineering flows.

8. Unsteady RANS simulations were also completed for a bridge-pier configuration consisting of two piers bents next to each other to study the interaction of adjacent pier bundles in a natural river. The geometry was obtained from an actual Chattahoochee River bridge near Cornelia, Georgia. The results were compared with experimental data and revealed a surprisingly strong interaction of the two pier bents, which manifested itself primarily in the turbulence statistics.

9. Unsteady RANS and hybrid URANS/LES computations were performed for flow past a bridge abutment. The computed flow patterns were found to be in remarkable agreement with the complex eddy motions observed in laboratory flow visualization experiments for the same configurations.

Teaching activities

1. Development of a new continuing education course in CFD Modeling for Complex Turbulent Flows. The course is being offered as part of the Georgia Tech continuing education program. It is aimed at consultant engineers, CFD code developers, and graduate students and it is designed to introduce the fundamentals of turbulence modeling and CFD for complex engineering flows.

2. A new graduate level CFD class has been developed and was offered for the first time during the Spring semester of 2001. The course is officially cross-listed as Civil Engineering and Mechanical engineering class but it is designed to attract students from various engineering disciplines within Georgia Tech, including, among others, chemical and biomedical engineering.

3. A new graduate level class in the area of Turbulence modeling for complex flows was developed. The objective of the course is to introduce the fundamental of statistical turbulence modeling and to incorporate our latest research developments in the graduate curriculum. The course was offered for the first time during the Spring semester of 2002.

4. A web site has been developed and will be online in the coming weeks. The web site is aimed at disseminating the results of our numerical modeling simulations for a broad range of engineering problems. Several images and video animations will be available, which can be used as educational supplements to both undergraduate and graduate fluid mechanics courses. The first draft of this web site can be found at: http://www.ce.gatech.edu/~fs30/GroupWebpage/Introduction.htm

The final version will be online in October 2004.

Findings: (See PDF version submitted by PI at the end of the report)

Training and Development:

All students and postdocs that have worked on this project have been trained to the latest advancements in computational fluid dynamics for complex engineering flows. They have developed unique skills that allow them to develop state-of-the-art computational algorithms, apply these algorithms to solve complex engineering flow problems, and analyze the rich physics of such flows.

Special emphasis was placed on involving undergraduate students in NSF funded research. Ms. Tahirih Lackey, whose thesis work was funded by this project and will be defending her PhD this Fall, was re-cruited in the group after taking the CEE undergraduate fluid mechanics class taught by the PI.

Outreach Activities:

I established an ASCE/EWRI task committee aimed at better informing practicing hydraulic engineers and consultants about the capabilities and limitations of computational fluid dynamics as a tool for solving complex engineering flow problems. The objectives and mandate of the committee are outlined in:

Sotiropoulos, F., and Wei, C. Y., 'New Task Committee on Advanced Environmental-Hydraulics Modeling,' ASCE Journal of Hydraulic Engineering, 127(1), pp. 3-4, 2001.

Journal Publications

Chrishochoides, A., Sotiropoulos, F., and Sturm, T. W., "Coherent Structures in Flat-Bed Bridge Abutment Flows: Experiments and CFD simulations", ASCE Journal of Hydraulic Engineering, p. 171, vol. 129, (2003). Published

Sotiropoulos, F., Webster, D. R., and Lackey, T. C., "Experiments on Lagrangian Transport in Steady Vortex Breakdown Bubbles in a Confined Swirling Flow", Journal of Fluid Mechanics
, p. 215, vol. 466, (2002). Published

I. Mezic and F. Sotiropoulos, "Ergodic Theory and Experimental Visualization of Invariant Sets in Chaotically Advected Flows", Physics of Fluids, p. 2235, vol. 14, (2002). Published

Ge, L., Jones, S. C., Sotiropoulos, F., Healy, T., and Yoganathan, A., "Numerical Simulation of Flow in Mechanical Heart Valves: Grid Resolution and Flow Symmetry", ASME Journal of Biomechanical Engineering, p. 709, vol. 125(5), (2003). Published

H. Tang, S. C. Jones, and F. Sotiropoulos, "An Overset Grid Method for 3D, Unsteady,Incompressible Flows", Journal of Computational Physics, p. 567, vol. 191(2), (2003). Published

A. Chrisohoides and F. Sotiropoulos, "Experimental Visualization of Lagrangian Coherent Structures in Aperiodic Flows", Physics of Fluids
, p. 25, vol. 15, (2003). Published

Sotiropoulos, F., Ventikos, Y., and Lackey, T. C., "Chaotic Advection is Stationary Vortex Breakdown Bubbles: Silnikov's Chaos and the Devil's Staircase", Journal of Fluid Mechanics, p. 257, vol. 444, (2001). Published

Jones, C. S., Sotiropoulos, F., and Amirtharajah, A., " Numerical Modeling of Helical static Mixers in Water Treatment", ASCE Journal of Environmental Engineering, p. 431, vol. 128, (2002). Published

Gilmanov, A., and Sotiropoulos, F., "A Hybrid Cartesian/Immersed Boundary Method for Simulating Flows with 3D Geometrically Complex Moving Bodies", Journal of Computational Physics, 2004, p. , vol. , ( ). Submitted

Gilmanov, A., Sotiropoulos, F., and Balaras, E., "A General Reconstruction Algorithm for Simulating Flows with Complex 3D Immersed Boundaries on Cartesian Grids", Journal of Computational Physics, 2003, p. 660, vol. 191(2), (2003). Published

Ge, L. and Sotiropoulos, F.,, "3D Unsteady RANS Modeling of Complex Hydraulic Engineering Flows. Part I: Numerical Model", ASCE Journal of Hydraulic Engineering, 2004, p. , vol. , ( ). Submitted

Ge, L., Lee, S, Sotiropoulos, F., and Sturm, T. W, "3D Unsteady RANS Modeling of Complex Hydraulic Engineering Flows. Part II: Model Validation and Flow Physics", ASCE Journal of Hydraulic Engineering, 2004, p. , vol. , ( ). Submitted

Lackey, T. C., and Sotiropoulos, F, "Numerical Simulation of Steady and Unsteady Free-Surface Flows Using Swallow-Water Equations", ASCE Journal of Hydraulic Engineering, p. , vol. , ( ). Preliminary accepted, pending final review

Paik, J., Ge, L., and Sotiropoulos, F., "Recent Progress in Simulating Complex 3D Shear Flows Using Unsteady Statistical Turbulence Models", Int. Journal of Heat and Fluid Flow, p. 513, vol. 25(3), (2004). Published

Paik, J., Sotiropoulos, F., and Sale, M. J., "Numerical Simulation of Swirling Flow in a Complex Hydro-Turbine Draft Tube Using Unsteady Statistical Turbulence Models", ASCE Journal of Hydraulic Engineering, 2004, p. , vol. , ( ). Submitted

Tang, H., and Sotiropoulos, F., "Fractional Step Artificial Compressibility Method for the Incompressible Navier-Stokes Equations", Computers and Fluids, 2004, p. , vol. , ( ). Submitted

Books or Other One-time Publications

Fotis Sotiropoulos, "Progress in Modeling 3-D Shear Flows Using RANS Equations and Advanced Turbulence Closures", (2001). Book chapter, Published
Editor(s): M. Rahman and G. Tzabiras
Collection: Calculation of Complex Turbulent Flows
Advances in Fluid Mechanics Series
Bibliography: Computational Mechanics Publications, Southampton, UK

F. Sotiropoulos, "Turbulence Modeling for Environmental Flows", (2004). Book, Accepted
Editor(s): John Wiley
Collection: Computational Fluid Mechanics: Applications in Environmental Hydraulics
Bibliography: To appear

Hansong Tang, "Numerical Simulation of 3D Unsteady Incompressible Flows in Complex Geometries", (2001). Thesis, Published
Bibliography: Ph.D. Thesis, School of Civil and Environmental Engineering, Georgia Institute of Technology, Atlanta, GA.

Tahirih Lackey, "Numerical Investigation of Chaotic Advection in Confined Swirling Flows", (2004). Thesis, Thesis completed and will be defended in Nov. 2004.
Bibliography: Ph.D. Thesis, School of Civil and Environmental Engineering, Georgia Institute of Technology, Atlanta, GA

Liang Ge, "Numerical Simulation of Complex 3D Turbulent Flows with Unsteady Coherent Structures", (2004). Book, Thesis completed, to be defended in Nov. 2004
Bibliography: Ph.D. Thesis, School of Civil and Environmental Engineering, Georgia Institute of Technology, Atlanta, GA

Web/Internet Site

URL(s):

Description:

Other Specific Products

Product Type: Ph.D. Thesis

Product Description:
The Ph.D. Thesis of Hansong Tang was funded by this project.

"Numerical Simulation of Unsteady Three Dimensional Flows in Complex Geometries," School of Civil and Environmental Engineering, Georgia Tech, October 2001.

Sharing Information:
Published Ph.D. thesis

Contributions

Contributions within Discipline:
The numerical computations of turbulent flow past a typical bridge abutment have provided the first insights into the enormous complexity of

such flows. Previously unknown flow features have been uncovered and their potential effect on scour processes has been discussed. These findings have: 1) underscored the need for detailed experimental studies; 2) motivated the development of novel experimental techniques (see below); and 3) demonstrated that the simplistic sediment transport models used widely today in engineering computations may not be capable of quantitatively accurate predictions of scour at bridge foundations.

The development of the domain decomposition method allows, for the first time, the simulation of real-life bridge sections using advanced unsteady RANS models on fine computational meshes. The method can simulate unsteady vortex shedding in arbitrarily complex geometrical configurations and will, thus, lead to new insights into the structure and dynamics of the scour inducing coherent vortices near bridge foundations.

The experimental technique we have developed for visualizing large-scale vortices in the vicinity of bridge foundations can provide unique insights into the dynamics of coherent structures in such flows. Both qualitative and quantitative (number of structures and shedding frequencies of various vortices) information can be extracted from this technique. Such information can be used along with traditional mean velocity and turbulence statistics measurements to validate CFD models.

We have developed and successfully validated the first advanced CFD model for carrying out unsteady RANS computations for real-life bridge foundations. Our model can simulate complex, multi-connected hydraulic structures in natural rivers using second-order accurate numerics and fine computational grids. The model resolves directly the dynamics of foundation induced coherent vortices and models the turbulent contributions via statistical turbulence models. The development of this model has now made it possible to tackle the modeling of scour-inducing sediment transport processes at a level of sophistication not previously possible--essentially all existing numerical models for simulating scour rely on simplistic (steady-state) hydrodynamic models. Such work is currently under way.

Contributions to Other Disciplines:

Chaotic mixing in complex vortical flows: Our flow solver and Lagrangian particle tracking algorithms developed with NSF support have been applied to elucidate the complex Lagrangian transport and stirring mechanisms in vortex breakdown bubbles. This work has led to new insights into the chaotic dynamics of such flows and could have significant implications in several industrial devices which rely on vortex breakdown to enhance mixing and transport. This work has already been published in the Journal of Fluid Mechanics.

Experimental Visualization of Chaotically Advected Flows: Our computational work on chaotic advection inspired the development of the first, non-intrusive, experimental technique for visualizing unmixed islands in chaotically advected swirling flows. With NSF funding, the technique was applied to provide the first experimental verification of our computational findings concerning the complex Lagrangian dynamics of vortex breakdown bubbles. This work has already been published in a series of papers in the Journal of Fluid Mechanics and Physics of Fluids.

Numerical Modeling of Flows in Prosthetic Mechanical Heart Valves: The domain decomposition method we have developed with NSF funding for
bridge foundation flows has been extended to simulate unsteady pulsatile flows in mechanical prosthetic heart valves. The first application of the method to simulate 3D flow patterns in a bileaflet valve have been reported in a paper, which was recently published in the ASME Journal of Biomechanical Engineering.

Unsteady modeling of flows in hydroturbine draft tubes: The domain decomposition methodology developed in this project was also extended to simulate swirling flows in real-life hydroturbine draft-tubes. A paper describing the method and its first application to a very complex draft-tube geometry has been recently submitted for publication in the ASCE J. of Hydraulic Engineering.

Simulation of a turbulent boundary layer on a concave wall: We carried out numerical simulations of turbulent flow past a concave wall using unsteady statistical turbulence models. Our work showed for the first time that such models can capture the onset of Taylor-Gortler rolls inside the centrifugally unstable boundary layer and, thus, account for the well-known dramatic effect of concave curvature on the structure of turbulence. The results of this work were reported in a recent (invited) paper in the Int. J. of Heat and Fluid Flow.

Experimental determination of coherence time scale in aperiodic mixing: The experimental technique we developed to visualize coherent structures near bridge abutments has formed the basis for a novel experimental approach for extracting the coherence time scale of Lagrangian coherence structures from experimental time series of light intensity in studies of mixing in aperiodic flows. To the best of our knowledge the technique we proposed for determining the coherence time scale is the first of its kind in this field. This work has already been published in Physics of Fluids.

CFD modeling of mixing in helical static mixers: The flow solver that has been developed with NSF funding has been modified to simulate laminar and turbulent flows within helical static mixers. Such devices are used in waste and water treatment plants as mixing devices. Our work has demonstrated, for the first time, the potential of advanced CFD methods as a powerful tool for elucidating the complex, vortical structures that control mixing in such flows. This work has already appeared in the ASCE J. of Environmental Engineering.

NSF funds from this award have also been used to support the development of a numerical method for simulating flows past swimming aquatic animals and flying insects. The method employs a novel sharp-interface approach to account for the interaction of the flexible, undulating fish body with the flow, which is based on a hybrid Cartesian/Immersed Boundary methodology. One paper describing the basic idea of the method has already been published while another paper is currently under review in the Journal of Computational Physics.

Contributions to Human Resource Development:

Contributions to Resources for Research and Education:

Contributions Beyond Science and Engineering:

Categories for which nothing is reported:

Contributions: To Any Human Resource Development

Contributions: To Any Resources for Research and Education

Contributions: To Any Beyond Science and Engineering

1999 ASCE Int. Water Resources Engineering Conf., **Seattle, WA, August 1999**

Prediction of Bridge Abutment Flows Using Advanced Turbulence Closures

Fotis Sotiropoulos, Antonis Chrisochoides, and Terry W. Sturm[*]

Abstract

A second-order accurate, finite-volume numerical model for solving the unsteady, three-dimensional Reynolds-averaged Navier-Stokes (RANS) equations is developed and applied to study bridge abutment flows. Calculations are performed for a typical abutment configuration and the computed solutions are shown to be in reasonable agreement with general trends observed in laboratory experiments. The simulations reveal the presence of highly three-dimensional, unsteady in-the-mean, separated flow regions both upstream and downstream of the abutment. These complexities underscore the need for comprehensive laboratory experiments to facilitate further modeling refinements.

1. Introduction

Scour of the streambed at bridge piers and abutments (collectively referred to as foundations) during flood events has resulted in more bridge failures than all other causes in recent history.[1] For this reason, bridge foundation scour has been the subject of intense research for well over 100 years.[2] Despite, however, a rapidly expanding body of literature on this topic, the mechanisms that govern the interaction of the approach flow with the foundation structure and the river bed and lead to scour are still not entirely understood.[2] Consequently, most existing approaches for estimating scour and remedies for ensuring the structural integrity of bridges during severe flooding rely on empirical correlations derived using dimensional analysis arguments with input from field and laboratory experiments.[3,4]

This state of affairs should be attributed to the unsteady, three-dimensional character of the flow phenomena that induce scour and their non-linear interaction with the mobile bed.[2] As the approach flow encounters a bridge foundation it rolls to form multiple large-scale vortices whose axes may be parallel (horseshoe or necklace-type vortices) or perpendicular (tornado and/or whirlpool-type vortices) to the bed. These unsteady large-scale flow structures in conjunction with a broad range of turbulent scales control the sediment transport in the vicinity of the foundation and are responsible for the growth of scour.[3,5] Prerequisite, therefore, for understanding the fundamental mechanisms of scour is to elucidate the nature of the foundation-induced vortices and their interaction with the river bed.[5] Most previous research efforts, however, focused on small-scale laboratory experiments that were aimed at developing empirical scour-depth equations rather than understanding the fluid mechanics of local scour. Field experiments can provide a more complete description of the scour process but are difficult and expensive to carry out since they have to be conducted during the actual flooding event.

The enormous complexity of the scour problem and the difficulties in obtaining the much needed physical insights from experiments alone, point to the need for a comprehensive three-pronged research strategy consisting of field measurements, small-scale laboratory studies, and three-dimensional numerical modeling.[2] In this study, we report recent progress toward the

---

[*] School of Civil and Environmental Engineering, Georgia Institute of Technology, Atlanta, GA 30332-0355

development of an advanced Computational Fluid Dynamics (CFD) model that can yield quantitatively accurate predictions of the three-dimensional vortical flow structures in the vicinity of bridge foundations. The model is based on the unsteady, three-dimensional Reynolds-averaged Navier-Stokes (RANS) equations closed with the k-ω turbulence closure.[6] Calculations are reported for a typical abutment configuration. The computed results are compared with available experimental measurements and interrogated in depth to elucidate the details of the complex, three-dimensional vortical structures that dominate the flow in the vicinity of the foundation.

2. The Numerical Method and Computational Details

The numerical method solves the unsteady, three-dimensional RANS equations in conjunction with the k-ω turbulence closure. The governing equations are formulated in generalized curvilinear coordinates and discretized on a non-staggered mesh via a conservative finite-volume scheme. The viscous terms in the mean flow and turbulence closure equations and the source terms in the turbulence closure equations are approximated using three-point, second-order accurate central differencing. The convective terms in the mean flow equations are discretized using central differencing with fourth-difference, third order, matrix-valued artificial dissipation terms for stability. The physical time derivative in the momentum and turbulence closure equations is discretized using a backward second-order accurate formula. The discrete equations are transformed into a pseudo-compressible, hyperbolic-like system by introducing pseudo or dual temporal derivatives of the pressure, velocity components, and turbulence quantities in the continuity, momentum, and turbulence closure equations, respectively. At every physical time step these pseudo-unsteady terms are driven to a prescribed small tolerance using a pointwise implicit, Runge-Kutta algorithm enhanced with local (dual) time-stepping, implicit residual smoothing and multigrid acceleration.[7]

The abutment geometry we simulate herein along with the curvilinear, body-fitted mesh we employ is shown in figure 1. The computational domain begins $10L_a$ (where L_a is the abutment length) upstream of the front face of the abutment and extends $20L_a$ downstream. The channel depth is set equal to $d=0.1L_a$—in this study the free-surface is treated as a flat rigid lid. The far field boundary is placed $3.5L_a$ from the channel sidewall where the abutment is located and it is treated as a plane of symmetry. At the inflow boundary, we specify fully-developed distributions for the mean axial velocity and the turbulence quantities obtained from a separate straight-duct calculation. At the outflow boundary we employ a convective boundary condition for the mean velocity components, which allows for flow structures to smoothly exit the solution domain. The pressure at all boundaries is obtained via linear extrapolation from the interior nodes. The calculations reported below were carried out for $Re=10^5$, based on the bulk velocity, U_b, and the abutment length. Finally, the computational grid was generated using an elliptic grid generation approach. It consists of 161x161x45 grid nodes ($1.16\text{x}10^6$ nodes) in the axial, horizontal, and vertical directions, respectively. The first grid nodes off all solid boundaries are located, almost everywhere, inside the laminar sublayer.

3. Description of the Laboratory Experiments

Experiments were conducted in a 4.2 m wide by 24.4 m long recirculating flume with an entrance tank containing flow stilling devices and a tailgate for control of the tailwater elevation and have been described previously[4]. Scour depths were measured as a function of discharge, sediment size, abutment length, and abutment shape for a compound channel cross-section constructed inside the flume at a fixed bed slope of 0.0022. Depth-averaged velocities at the bridge approach section and near the abutment face as well as water surface profiles were measured for the fixed-bed case to simulate conditions at the beginning of scour. The location of the stagnation point on the upstream side of the abutment was measured using a dye tracer released from the bridge approach section.

The compound channel section consisted of an asymmetric floodplain of width 3.66 m next to a trapezoidal main channel having a side slope of 2.5:1 on one side and a vertical wall on the other. A lean concrete mix that utilized the movable-bed sediment (d_{50} = 3.3 mm) as aggregate was poured to form the fixed-bed compound channel section. The protruding sediment grains in the bed surfaces resulted in fully-rough turbulent flow. After a series of velocity and water-surface profile measurements were completed for the fixed-bed channel, a 6-m long section midway along the flume was made into a moveable bed. Both vertical-wall and spill-through bridge abutments were located 9.8 m downstream of the flume entrance and 11.6 m upstream of the tailgate. The relative abutment length ratios with respect to the total channel width B and with respect to the downstream normal depth y_{f0}, which was set as the downstream boundary condition, are given in Table 1 for those experimental runs most nearly comparable to the numerical computation case. Also shown in the table are the Reynolds number ranges (based on the approach floodplain velocity and the abutment length) and measured reattachment lengths of the upstream recirculating zone. Scour measurements were made for several discharges at each of the abutment lengths given in Table 1. Scour was allowed to continue for 24 to 36 hours. After equilibrium had been reached, the bed elevations throughout the scour area were measured with a point gauge resulting in an uncertainty in scour depths of about 1.0 mm.

4. Results and Discussion

Although the present simulations have assumed fixed bed, it is useful to compare observed in the laboratory shapes of scour holes with distributions of flow quantities that are typically used in sediment transport models to define thresholds for the initiation of grain motion. Figure 2 compares calculated contours of bed shear velocity and vertical mean velocity component with the scour patterns observed in the laboratory for an abutment geometrically similar (albeit not identical) to the one used in the simulations. The shape, location, and size of the measured scour hole correlate well with the calculated pocket of high shear velocity. Interestingly, the calculations also indicate that within this pocket of high shear stress there is a region of negative vertical velocity. For continuity to be satisfied, a vertically downward motion along the abutment wall must be accompanied by a horizontal motion away from the abutment along the bed, thus, setting up a counter-clockwise secondary motion—the calculated sense of rotation of the secondary motion is in agreement with the experimental observations summarized in Melville.[3] This finding suggests that the formation of the scour hole is due to the combined effects of high bed shear stress, which initiates sediment motion, and the counter-clockwise secondary motion that sweeps bed material away from the abutment.

Let us now focus our attention on the three-dimensional structure of the flow in the vicinity of the abutment. Figure 3 shows calculated instantaneous streamlines at the surface and near the bed (limiting streamlines) in the vicinity of the upstream face of the foundation. Previous experiments[3] as well as our own laboratory studies[4] have documented the existence of a recirculating eddy in the junction region between the abutment and the sidewall on the upstream side. As seen in figure 3, our simulations reproduce this feature but further reveal that the mean flow in this region is highly three-dimensional and unsteady. Rather than a single recirculating eddy, the flow actually consists of a series of counter-rotating eddies. New eddies are generated continuously, due to the complex interaction between existing eddies, and between the eddies and the surrounding walls. They merge to form larger eddies, which in turn break up into smaller eddies and so on. The near-bed limiting streamline plots in figure 3 indicate that there is no direct correspondence between the various singular points in the surface and near-bed streamlines. This feature suggests that the multiple surface eddies are not the footprints of vertical tornado-like structures that terminate at the bed. To clarify their origin, we plot limiting streamlines along the sidewall and the front face of the abutment (figure 4) and instantaneous 3-D particle trajectories in this region (figure 5). The spiral focus of the limiting streamlines along the vertical sidewall, seen in figure 4a, suggests the presence of an intense vortical structure whose axis is parallel to the bed. The presence of this vortex is clearly visible in the 3-D particle trajectories in figure 5. It appears, therefore, that the flow in this upstream corner is dominated by an intricate web of vortices, with axes perpendicular and parallel to the bed, which originate both from the bed and the vertical sidewall, and which interact with each other in a very complex, highly unsteady manner, giving rise to the multiple eddy patterns observed at the free-surface.

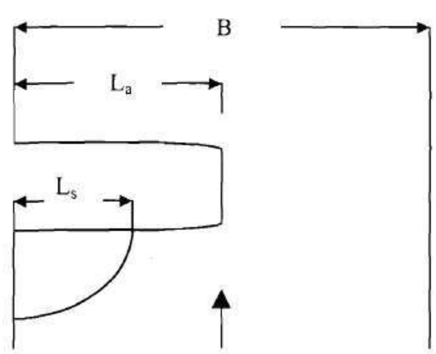
The limiting streamline plot along the upstream face of the abutment (see figure 4b) is also very revealing. Notice that the near-wall flow originating both at the surface and near the bed appears to converge along a well defined line of separation located just above the bed. This convergence line is the footprint on the abutment surface of the primary abutment vortex, which wraps around the foundation and is responsible for initiating and sustaining the scouring process. The formation of this vortex and the associated strong downward sweeping motion of the near-wall flow around the abutment edge are clearly visible in figure 5a as well.

To quantify the turbulence intensities in the vicinity of the abutment, we plot in figure 6 instantaneous contours of turbulence kinetic energy at various depths. The pocket of high turbulence kinetic energy that wraps around the abutment is induced by the intense mean velocity gradients in this region—due to the area contraction, the flow around the abutment accelerates and a shear layer develops between the fast moving outer flow and the very slow recirculating flow downstream of the abutment (see figure 7 below). What is rather surprising, however, is the presence of small pockets of very high levels of turbulence kinetic energy within the upstream corner recirculating zone. These pockets are highly three-dimensional and their location and intensity vary continuously in time. In fact for the instant depicted in figure 6, the largest values of k do not occur at the edge of the shear layer but in a small pocket at the front face of the abutment at approximately mid-depth. This intriguing finding, which suggests that the upstream recirculating zone is not as "quiescent" as thought earlier,[3] should be attributed to the complex interaction of the various vortical structures in this region with the surrounding walls.

The structure of the flow downstream of the abutment is depicted in figure 7. Our simulations reveal a massively separated wake consisting of several large-scale eddies and extending up 5 to 6 abutment lengths downstream. There are two main eddies that remain relatively stable in time (within the simulated time interval): a clockwise rotating eddy attached to the downstream face of the abutment; and a counter-clockwise eddy at the downstream end of the recirculating flow region. New eddies are shed from the abutment, convected downstream along the shear layer and merge with the large counter-rotating eddy. An additional eddy forms along the sidewall due to the very complex interaction between the shear layer and the sidewall. As indicated by the wavy structure of the iso-vorticity contours, the shear layer undergoes intense temporal oscillations in the horizontal plane. The downstream end of the elongated zone of high negative vorticity rolls up periodically, sending a pocket of negative vorticity plunging onto the sidewall. This pocket interacts with the sidewall, causing the injection of a tongue of positive vorticity from the near-wall region into the recirculating zone and the formation of the new eddy as shown in the sequence in figure 7.

Finally, we should point out that we do not have as yet experimental measurements of sufficient detail to confirm the existence of the very complex flow patterns revealed by our calculations. Some quantitative comparisons that support the accuracy of our computational findings are summarized in Table 1 below. This table compares observed laboratory reattachment lengths for the upstream recirculating zone (see sketch for definitions) with computed values. Given the fact that simulated abutment geometry is not exactly identical to any of the laboratory configurations and the uncertainties in measuring the reattachment lengths in an unsteady flow, the computed results are in reasonable overall agreement with the laboratory observations.

	Configuration	L_a/B	L_a/y_{fo}	Re	L_s/L_a
Experimental results	Vertical wall/Square edges	0.22	24.2-13.6	$2\text{-}2.5\times10^5$	0.5-0.6
	Vertical wall/Square edges	0.44	53.5-32.3	$3\text{-}4.2\times10^5$	0.4-0.5
	Spill-through	0.32	35.6-20.0	$3\text{-}3.5\times10^5$	0.39-0.54
Numerical results	Vertical wall/Rounded edges	0.29	10	1.0×10^5	0.55-0.65



y_{fo}: depth downstream of the abutment; *Re*: Reynolds number based on L_a and approach velocity

Table 1: Laboratory and numerical configurations and comparisons of the size of the upstream recirculation zone

5. Summary and Conclusions

A numerical model was developed for solving the unsteady, three-dimensional incompressible RANS equations in generalized curvilinear coordinates. The model was applied to simulate flow past a typical abutment geometry. The computed solutions were found to be in good qualitative agreement with laboratory observations. Especially encouraging is the fact that the calculated pocket of maximum bed shear stress correlates well with the observed laboratory scour hole patterns. The simulations suggest that the growth of the scour hole is enhanced by a downward vertical velocity component in the vicinity of the pocket of maximum bed shear stress, which sets up a counter-clockwise secondary motion that sweeps bed material away from the abutment. Detailed interrogation of the computed solutions revealed that the mean flow in the vicinity of the foundation is highly three-dimensional and unsteady. The region of "quiescent" recirculating flow at the upstream face of the abutment is extremely complex as it is dominated by multiple vortices and regions of intense production of turbulence kinetic energy. Downstream of the abutment, there is a large region of separated flow, consisting of multiple large scale eddies, that extends several abutment lengths downstream.

It is evident from the present numerical simulations, that the descriptions of abutment flows obtained by advanced CFD computations are considerably more complicated than those derived by interpreting the results of existing laboratory studies.[3] Future experimental studies, therefore, should focus on obtaining detailed three-dimensional mean flow and turbulence statistics measurements which will be used to validate and refine the predictive capabilities of advanced CFD models. This is a critical prerequisite before the existing numerical methodologies can be extended to develop reliable numerical models of the scouring process.

The first author acknowledges the financial support of the National Science Foundation in the form of a CAREER award (CMS-9875691).

References

[1] Shirole, A. M., and Holt, R. C. (1991), "Planning for a Comprehensive Bridge Safety Assurance Program," *Transportation Research Record* No. 1290, Volume 1, 39-50.

[2] Parola, A. C., Hagerty, D. J., Mueller, D. S., Melville, B. W., Parker, G., and Usher, J. S. (1997b), "The Need for Research on Scour at Bridge Crossings," Proc. Of XXVII IAHR Congress, San Francisco, CA, August 10-15, 1997, pp. 124-129.

[3] Melville, B. W. (1997), "Pier and Abutment Scour: Integrated Approach," *ASCE J. Hydr. Eng.*, 123 (2), pp. 125-136.

[4] Sturm, T.W. and Chrisochoides, A. (1998), "Abutment Scour in Compound Channels for Variable Setbacks," *Proc. Int. Water Resources Eng. Conference*, ASCE, Reston, VA, Vol. 1, pp. 174-179.

[5] Dargahi, B. (1990), "Controlling Mechanism of Local Scour," *ASCE J. of Hydr. Eng.*, 116(10), pp. 1197-1214.

[6] Wilcox, D. C., Reassessment of the Scale Determining Equation for Advanced Turbulence Models, *AIAA J.*, **26**(11), pp. 1299-1310, 1988.

[7] Sotiropoulos, F., and Ventikos, Y. (1998), "Transition from Bubble Vortex Breakdown to a Columnar Vortex in a Closed Cylinder with a Rotating Lid," *Int. J. of Heat and Fluid Flow* 19, pp. 446-458.

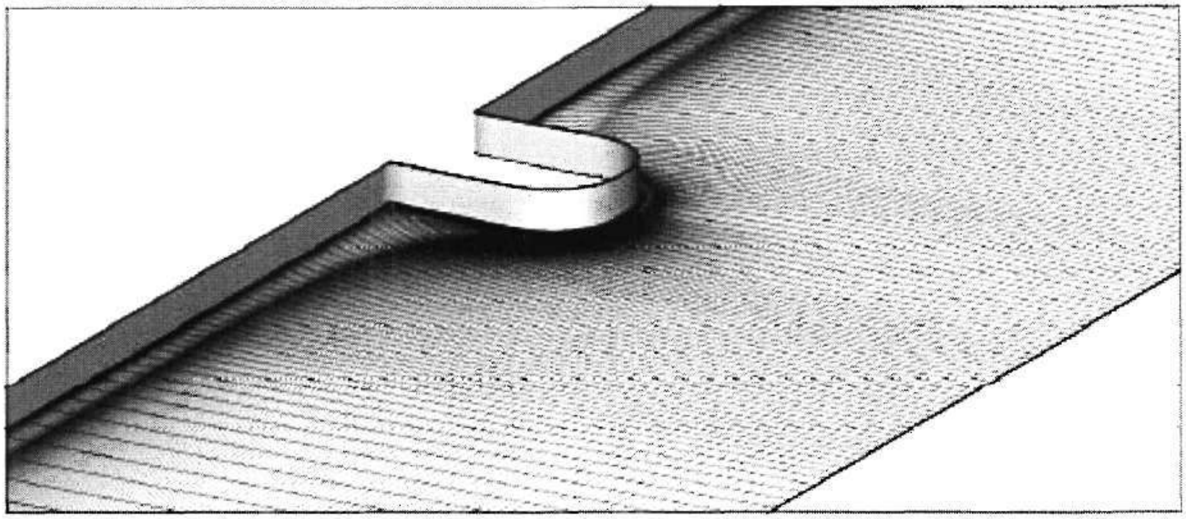


Figure 1. Simulated abutment geometry and computational mesh

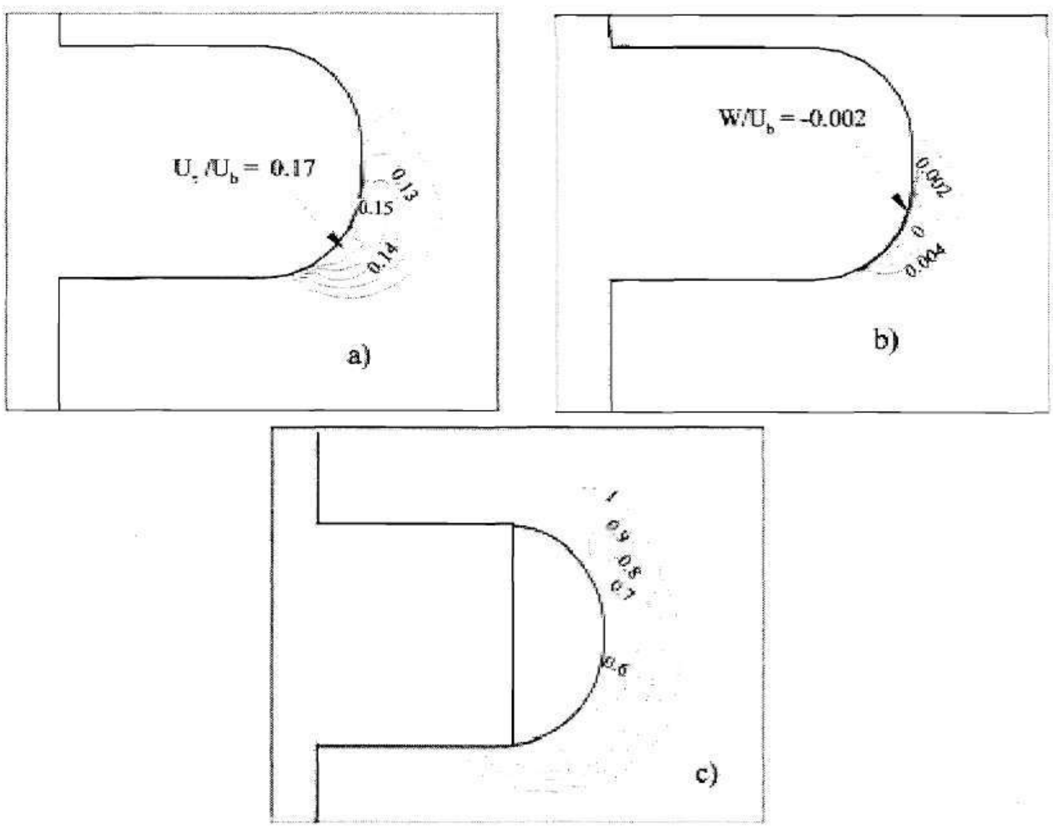


Figure 2. a) Computed contours of bed shear velocity; b) Computed contours of mean vertical velocity near the bed
c) Measured bed elevation contours (in feet) for a spill-through abutment (see Table)